The information given should only be used as a guideline or starting point for feed rate selection. Your actual feeds and speeds will vary widely as a result of "contributing factors" such as machine rigidity, horsepower, collet condition, spindle integrity, part clamping, hold down and many other factors. Generally speaking, solid carbide spiral tooling will perform better (i.e. longer life, less tool breakage) at faster feed rates. We recommend selecting a "starting point" feed rate and increasing that feed rate until part finish becomes undesirable or other limiting factors become evident.

Chip Load

The chip load is a measurement of the thickness of material removed by each cutting edge during a cut. This is a valuable piece of information which can then be used to calculate new set ups.

Calculations are as follows: Chip Load = Feed Rate (inches per minute) / (RPM x number of flutes)
Example: Chip Load = 500 inches per minute / (15,000 RPM x 2 flutes) Chip Load = .017"

Chip loads are based on material thickness of average size for cutting edge length of tool. These recommendations do not apply to thicker material or tools with long cutting edge lengths. These chiploads are only a recommended starting point and may not accomondate all circumstances. Therefore, tooling damage may still occur and use of this chart does not warranty against tool breakage.

We would strongly encourage you to consult us directly on new tool applications. Our staff would be happy to discuss any technical questions by phone or email.

Chip Load Chart

Tool Diameter	Hard Wood	Softwood Plywood	MDF/Particle Board	High Pressure Laminate **	Phenolic **
1/8"	.003" - .005"	.004" - .006"	.004" - .007"	.003" - .005"	.004" - .005"
1/4"	.009" - .011"	.011" - .013"	.013" - .016"	.009" - .012"	.011" - .012"
3/8"	.015" - .018"	.017" - .020"	.020" - .023"	.015" - .018"	.017" - .018"
1/2" & up	.019" - .021"	.021" - .023"	.025" - .027"	.023" - .025"	.024" - .026"

**Recommended RPM 9-10,000

Tool Diameter	Hard Plastic	Soft Plastic	Solid Surface	Acrylic	Aluminum
1/8"	.002" - .004"	.003" - .006"	.002" - .004"	.003" - .005"	.003" - .004"
1/4"	.006" - .009"	.007" - .010"	.006" - .009"	.008" - .010"	.005" - .007"
3/8"	.008" - .010"	.010" - .012"	.008" - .010"	.010" - .012"	.006" - .008"
1/2" & up	.010" - .012"	.012" - .016"	.010" - .012"	.012" - .015"	.008" - .010"

Other Valuable Formulas:

Feed Rate = RPM x number of flutes x chip load
RPM = feed rate/(number of flutes x chipload)
Metric Conversion: Divide inches per minute by 39.374 (ex. 300 inches per minute divided by 39.374 = 7.62 meters per minute)

RPM Selection - the general operating RPM for tooling contained in this catalog is between 10,000 and 20,000 revolutions per minute. Usually the higher the RPM, the better surface finish becomes. However, the higher the RPM, the higher the friction generated between the tool and the work piece. This friction is what creates the mechanical wear on the cutting edge. Your goal is to select the lowest RPM possible for each application.